

Book Name: *Part Modeling User's Guide*

Starting Out in Part Mode

This chapter provides an overview of the part building process.

Topic

Overview

Combining Features into Parts

Entering Part Mode

Setting Up the Part

Using the Model Tree Tool in Part Mode

Creating and Redefining Features

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Overview

Pro/ENGINEER provides both basic functionality and optional modules. Together, these make up the Pro/ENGINEER family of products. The following table lists the key functions for Pro/ENGINEER, Pro/FEATURE, Pro/INTERFACE, Pro/SURFACE, and Pro/FEATURE for BODY ENGINEERING. Use the Support Info option in the Misc menu to identify the optional modules available on your system.

If you do not have the appropriate license to perform a specific set of functions, you may need to use a different command to start Pro/ENGINEER, or you might be able to "float" the necessary options to your working session. For more information on floating options, see the Floating Modules section in Miscellaneous Functions in the *Fundamentals* manual.

If you have...	You can...
Pro/ENGINEER	<p>Create extruded and revolved protrusions, cuts, and slots (solid only).</p> <p>Create constant section sweeps, parallel blends, and swept blends.</p> <p>Create holes, shafts, chamfers, necks, flanges, ribs, and ears.</p> <p>Create rounds.</p> <p>Create regular drafts, local pushes, patches, and radius</p>

	<p>domes.</p> <p>Create sketched cosmetic features.</p> <p>Create datum planes, axes, points, curves, coordinate systems, and graphs.</p> <p>Create model cross-sections, and reference dimensions.</p> <p>Modify, delete, insert, pattern, suppress, resume, reroute, redefine, and reorder features; and make them read-only.</p>
Pro/FEATURE	<p>Create thin features.</p> <p>Create variable section sweeps for solids.</p> <p>Create rotational and general blends. Use IGES files to create blend sections. Create regular and projected section blends.</p> <p>Create a variety of drafts.</p> <p>Create section domes, offsets, toroidal bends, and lips.</p> <p>Create shells.</p> <p>Create pipes.</p> <p>Create grooves and threads.</p> <p>Create merge and cutout features in Assembly mode.</p> <p>Create user-defined features and local groups.</p> <p>Copy and mirror features.</p>
Pro/INTERFACE	<p>Import geometry from other CAD systems as solid, surface, or wireframe data.</p> <p>Incorporate the imported geometry into the existing Pro/ENGINEER model.</p>
Pro/SURFACE	<p>Create surface features.</p> <p>Create rounds using surfaces.</p> <p>Create cross-sections of surface features.</p> <p>Create surface models by translating, rotating, and mirroring surface features.</p> <p>Replace part surfaces with surface features.</p> <p>Create blends from files.</p> <p>Create blends between sections and surfaces, and between two surfaces.</p> <p>Create surfaces from a set of boundary curves in one or two directions.</p> <p>Create variable section sweeps for surface features.</p> <p>Create drafts and area offsets on a surface.</p> <p>Merge (join or intersect) surfaces together.</p> <p>Trim or extend surfaces.</p> <p>Create a new surface by offsetting or copying an existing one.</p> <p>Use an area graph to control a swept blend.</p>
Pro/FEATURE for BODY ENGINEERING	<p>Create draft offsets.</p> <p>Create perimeter dimensions in Sketcher.</p> <p>Control perimeter of a swept blend between its sections.</p>

License Restrictions

If a model or its features were created using an optional Pro/ENGINEER module for which you do not have a current license, there are limitations on what you can do with the model.

Without a license to a module, you can do the following:

- Modify the dimensions of features created with the missing module.
- Delete features created with the missing module.
- Suppress, reorder, resume, and insert features before the features from the absent module.
- Select imported geometry.

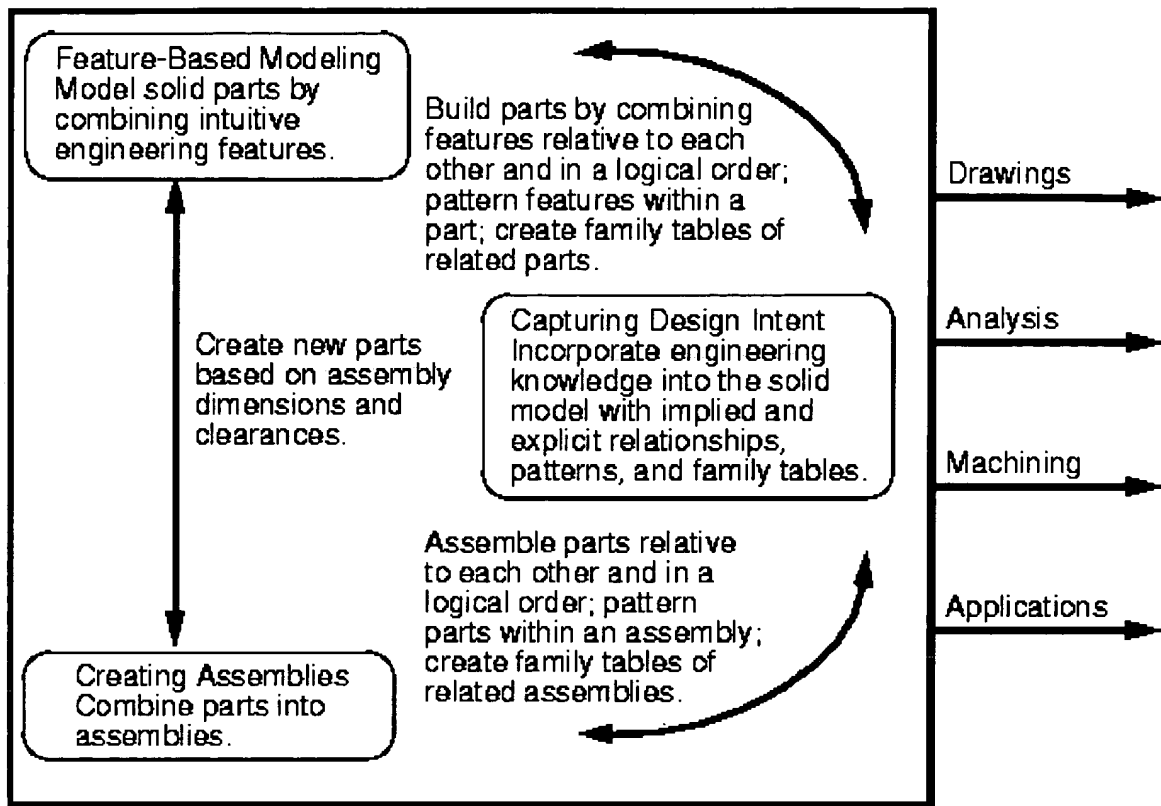
Without a license to a module, you cannot do the following:

- Create new features that require the absent module.
- Redefine features created with the absent module.

The Modeling Process

The following figure illustrates the modeling process.

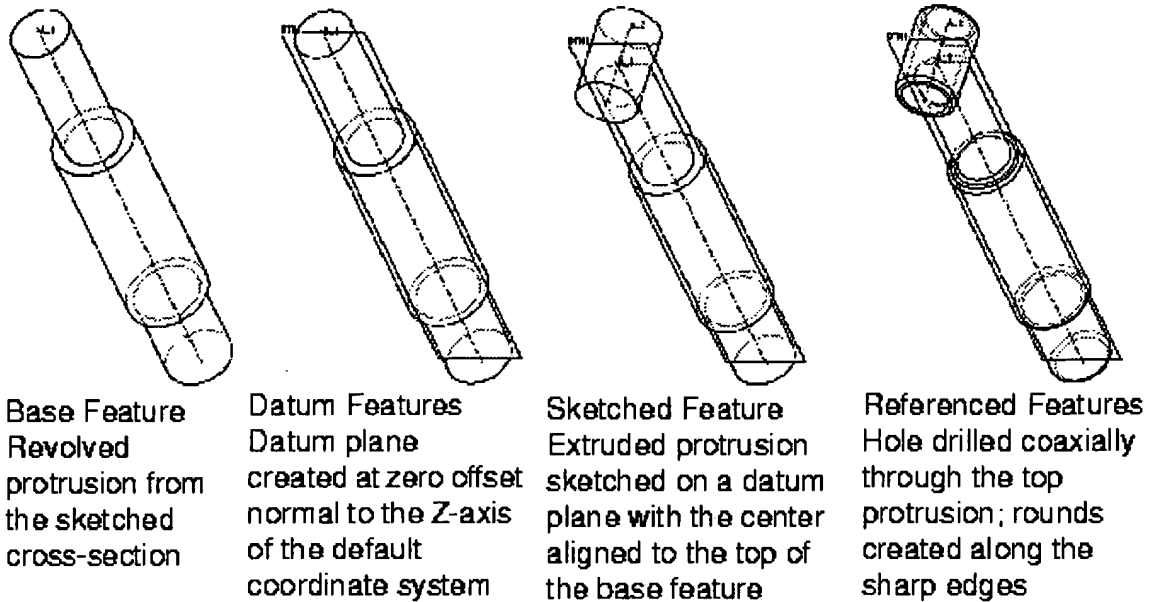
Parametric Solid Model



Combining Features into Parts

The rest of this chapter describes the concept of building parts. The various types of Pro/ENGINEER features are used as building blocks in the progressive creation of solid parts. Certain features, by necessity, precede others in the design process. The features that follow rely on the previously defined features for dimensional and geometric references. The progressive design of features can create relationships between features already in the design and subsequent features in the design that reference them. This is known as the *parent-child relationship*. The following figure illustrates the progressive design of features.

Progressive Design of Features



The parent-child relationship is one of the most powerful aspects of Pro/ENGINEER. When a parent feature is modified, its children are automatically recreated to reflect the changes in the geometry of the parent feature. It is therefore essential to reference feature dimensions so design modifications are correctly propagated throughout the model. Because children reference parents, features can exist without children, but children cannot exist without their parents.

For a description of parent-child relationships in the assembly modeling process, see the *Assembly Modeling User's Guide*.

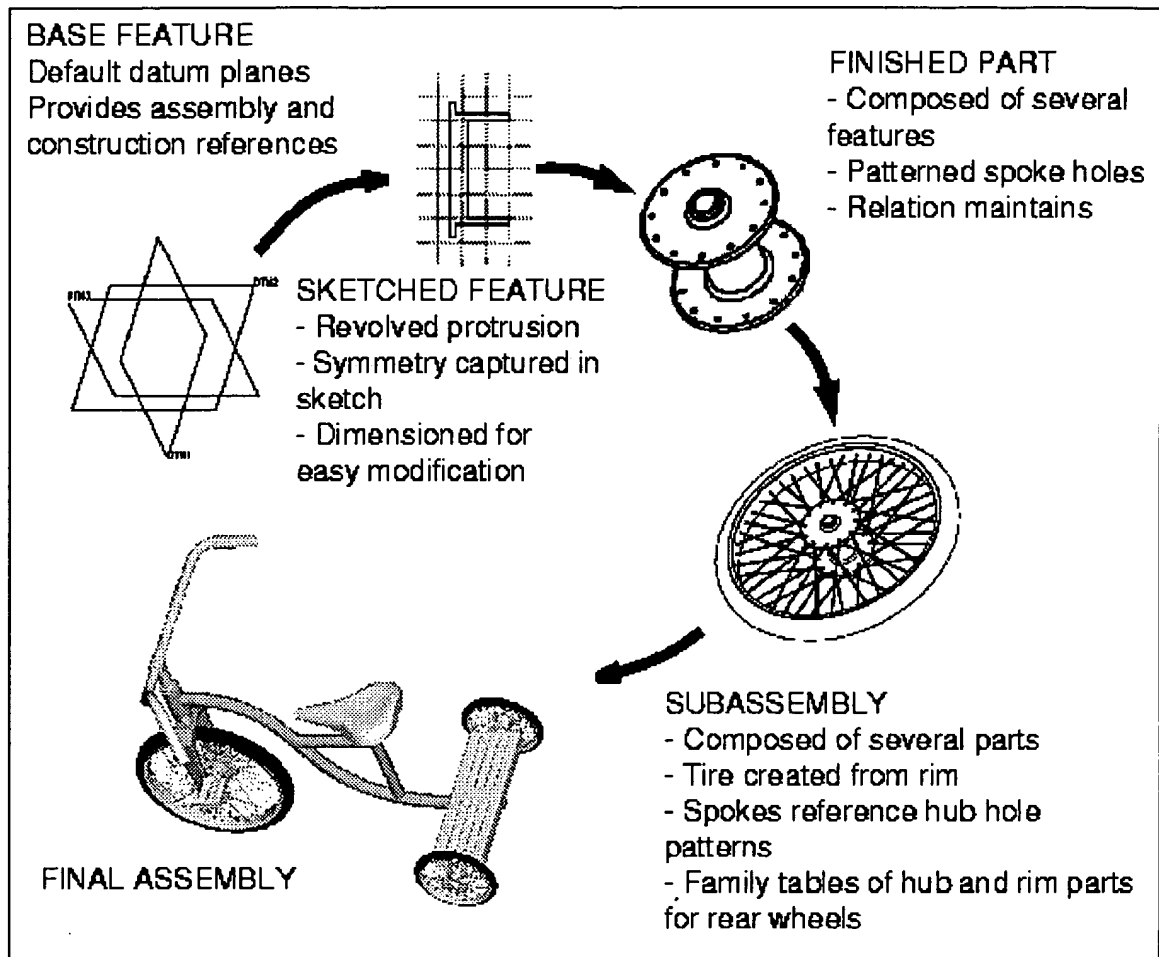
Putting It All Together

The modeling task is to incorporate the features and parts of a complex design into subassemblies and, ultimately, into a final assembly. At the same time, you must properly capture the design intent to provide flexibility in modification. Each Pro/ENGINEER parametric model is a careful synthesis of physical and intellectual design.

This process can also go in a top-down direction, breaking down a final assembly into subassemblies, parts, and features.

The following figure illustrates the design process from the base feature to the final assembly.

The Design Process



Entering Part Mode

To create a part, you must first enter Part mode. To do this, choose Part from the Mode menu. Pro/ENGINEER then displays the EnterPart menu. You can then create or retrieve a part by selecting the appropriate menu option.

The EnterPart menu options are as follows:

- **Create**-Create a new part.
- **Retrieve**-Retrieve a previously created part.
- **List**-List all the parts in the current directory.
- **Import**-Import a data file (IGES, SET, VDA, and so on) into an empty part. See the Interface Guide for information on importing files.
- **Export**-Export a specified part in the VRML format.
- **Search/Retr**-Search for and retrieve objects from the current directory, or from directories at different levels in the directory tree.

For more information on these options, see [Namelist Menus](#) in [Basic Information](#) in the *Fundamentals* manual.

Setting Up the Part

The first options available for a new part are Feature and Set Up. Before you start to create features, add relations, assign or generate mass properties, and so on, you should always use Set Up, then Units to make sure the unit value for the part is correct. This is important because certain aspects of the part cannot be modified appropriately if, for example, you decide to change the units of the part from inches to centimeters.

If you change the units of a part, Pro/ENGINEER automatically scales the parameters in a relation. However, any constant values used in the part might have to be changed manually. Non-parametric features (cosmetic and imported geometry) with non-modifiable dimension values will *not* be scaled.

You can set up layers and place items on them so you can blank selected layers, or make selections by layers. For information on layers, see [Layer Functions](#) in the *Fundamentals* manual.

Creating a Model Grid

You can create a three-dimensional grid for the model using the Part Setup menu. You can view this grid in Drawing mode only, where it can be displayed for a selected view or for an entire sheet.

How to create a model grid

1. Choose **Grid** from the PART SETUP menu.
2. The system displays the MODEL GRID menu, which lists the following options:
 - **Set Origin**-Specify the origin of the grid and select an existing coordinate system. If the grid currently has a set origin, you can change the origin by picking another coordinate system.
 - **Delete Grid**-Remove the grid.
 - **Grid Spacing**-Specify a spacing different from the spacing set by the "model_grid_spacing" option in the configuration file. The grid spacing is measured from the coordinate system you selected when you defined the grid origin.

Choose Set Origin.

3. The MODEL GRID menu displays a submenu with the **Grid On** and **Grid Off** options. The **Grid On** option allows you to display the grid if the model is in a valid orientation (that is, when one of the coordinate planes of the origin coordinate system is parallel to the screen).

The following configuration file options allow you to control the model grid:

- *solid_grid_neg_prefix*-Allows you to specify the prefix to use for negative numbers in the balloon capture of the grid. The system default symbol is "-".

- *solid_grid_num_dig*-Controls the number of decimal places displayed in the balloon capture. The system default value is 0 (that is, only integers will be displayed).

If you attempt to delete the coordinate system that acts as the grid origin, the system notifies you that doing so will clear the grid definition and asks for confirmation.

Using the Model Tree Tool in Part Mode

In Part mode, you can access some of the common feature operations through the Model Tree tool. These shortcuts enable you to do the following:

- Open a pop-up menu to access most frequently used feature operations (see a procedure below).
- Edit simplified representations of the model by excluding or including features through the Model Tree (this method is described in [Using the Model Tree to Edit a Simplified Representation](#)).

Additionally, you can use the Model Tree to create parameters and set their values (for details, see the *Fundamentals* manual).

How to use shortcuts through the Model Tree

1. With the left mouse button, select a feature in the Model Tree.
2. Press the right mouse button to open a pop-up menu. Note that the pop-up menu lists only commands that are valid for the selected feature; the menu on the right of the screen also restricts the choice of options in the pop-up menu.

If you are in the PART menu, the pop-up menu lists the following commands:

- **Modify**-Access the MODIFY menu to modify the selected feature.
- **Pattern**-Access the standard pattern functionality.
- **Delete**-Delete the selected feature. If the feature has children, the system highlights children in the Model Tree and in the model, and asks you for a confirmation to delete them with the selected feature.
- **Del Pattern**-Delete a feature pattern.
- **Ungroup**-Ungroup a group of features.
- **Unpattern**-Unpattern a patterned group.
- **Suppress**-Suppress the selected feature. If the feature has children, the system highlights children in the Model Tree and in the model, and asks you for a confirmation to suppress them with the selected feature. Once you suppress the feature, its status appears as *Suppressed* in the Status column of the Mode Tree.
- **Resume**-Resume the selected feature.

- **Redefine**-Access the feature creation environment so you can redefine feature elements.
- **Reroute**-Access the REROUTE menu to reroute the selected feature.
- **Info**-Access feature or model information by selecting **Feat Info** or **Model Info** from the submenu.

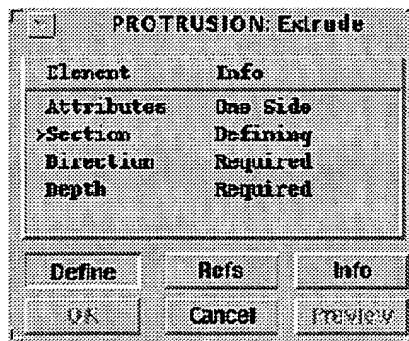
If you are in the Feat menu, the pop-up menu lists the Modify and Info options. The Info option lets you choose Feat Info or Model Info.

If you are in any menu under the Feat menu, the pop-up menu lists the Info option with Feat Info and Model Info.

For a top level part, you can create a new feature, or choose the Info option to access Model Info.

Creating and Redefining Features

When you create or redefine features that have elements, Pro/ENGINEER displays a dialog box that lists the elements of the feature and their current status. The following figure illustrates a sample dialog box.



The title of the dialog box describes the feature and subtype of the feature you are creating or modifying. For example, when you are creating an edge chamfer, Pro/ENGINEER displays the dialog box named *PROTRUSION: Extrude*.

When you are creating a feature, the system prompts you to select any optional elements (see [Selecting Elements](#)). When you have finished, select the Define button to start defining the feature. If the feature does not contain any optional elements, Pro/ENGINEER automatically initiates the definition process. Once initiated, the definition process continues until you have defined all the required elements. If you select Quit, control returns to the dialog box.

Note that while the dialog box is active, you still have access to several asynchronous options in the Main menu, including Environment, Misc, Info, and View.

The dialog box contains an element list and action buttons, which are described in the following sections.

Dialog Box Element List

The element list is a tabulated list box that contains two columns:

- *Element*-The element name (such as Scheme, Ref Surface, and Edge Refs).
- *Info*-The current state of the element. The initial status of an element is Undefined. During subsequent definition actions, the status changes to indicate that it has been, or will be, defined or changed. The possible status values are as follows:
 - *Undefined*-The element has not yet been defined. This is the initial state for an element, unless you have preassigned a default value.
 - *Defined*-The element has been completely defined. The system displays this status during the definition action once the specific element has been defined. This happens only if there is no helpful information to put there. Otherwise, the system displays the helpful information there, such as "Depth = 2.0".
 - *Defining*-The definition of the element is currently in process.
 - *To Define*-The definition of this element will occur later during the definition action.
 - *Changing*-The redefinition of the specified element is currently in process.
 - *To Change*-The redefinition of this element will occur later during the definition action.

The Info field also specifies whether an element is required or optional. If the element is required, you must specify the element to complete the feature. If the element is optional, you can include the element by selecting it.

Dialog Box Command Buttons

The command buttons in the dialog box are as follows:

- *Define*-Defines or redefines all the undefined required and selected elements, working from the top of the element list to the bottom. The system always tries to define any undefined but required elements. Once you have selected the button, the status for each marked element is updated accordingly (such as being changed from "Defined" to "To change").

The system changes the status of the element being processed and places a ">" character to the left of the element.

Pro/ENGINEER displays the menus for the element currently being processed to the right of the graphics window and displays any messages or prompts in the Message Window.

Once the element has been defined, the system updates its status to Defined and starts to process the next element.

When you have defined all of the elements, control returns to the dialog box. The system displays a message stating that all the elements have been defined.

- *Refs*-If available, this option displays the references for the selected elements, working from the top of the element list to the bottom.

You can advance through each of the references for the selected element, go to the next selected element, or quit using the menus displayed by Pro/ENGINEER.

When you select the Show Refs button in the dialog box, the system displays the Show Refs menu. The menu lists the following options:

- **Next**-Go to the next item.
- **Prev**-Go to the previous item.
- **Info**-Get information on the current reference.

When the action has been completed, control returns to the dialog box.

Note:

The Show Refs option is not available for features created prior to Release 16.0.

- *Info*-Displays the information on the feature. For UDFs, this button is called "UDF Info"; for groups, this button is called "Group Info."
- *Refs*-Displays the references for the entire feature. For UDFs, this button is called "UDF Refs"; for groups, this button is called "Group Refs".
- *OK*-Completes feature creation and closes the dialog box.
- *Cancel*-After confirmation, this cancels feature creation, or discards any changes that have been made during feature redefinition, and closes the dialog box.

You can use this button at any time during feature creation or redefinition to immediately quit the process.

- *Preview*-Displays the feature geometry as it will be built, with the exception of UDFs and groups.

When you select the Preview button in the dialog box, the system displays a preview of the new feature. If you perform any view or information operations, the feature geometry remains in the part. It disappears if you abort the feature creation or redefinition. If you go to redefine elements, the system displays the geometry until the first repaint.

If the feature has geometry checks, Pro/ENGINEER asks if you want to see more information. If you respond "yes", the system displays the Show Errors menu. See Geometry Checking in the Chapter Regenerating the Part for more information on the Show Errors menu.

- *Resolve*-This button appears only if the feature fails after you choose Preview or OK from the dialog box. By pressing this button, you to enter the "fix model" environment. For more information, see Changing Part Accuracy.

Selecting Elements

Pro/ENGINEER displays a ">" character to the left of the current element. You can move the cursor up and down the element list using *arrow keys*.

Double-clicking on an element automatically selects that element and selects the Define button.

The following table describes how to select and unselect elements in the element list.

Action	Result
Click anywhere on the line of the element.	Selects that element and removes the selection from any previously selected elements.
Select an element with the <Ctrl> key pressed down.	Selects that element without unselecting other elements.
Drag the mouse with the left button pressed over the elements.	Selects all the elements the cursor passes over.
Press the <Ctrl> key and click and drag the cursor over the elements.	Appends the selected elements to the current selection.
Click on an element with the <Shift> key pressed.	Selects that element and all the elements between it and the last selected element.
Click on a highlighted element.	Removes the selection from that element.

Configuration Options for Dialog Boxes

There are two configuration options that enable you to control the process of feature creation through the dialog box:

- "feature_create_auto_begin"-Determines whether the Define button in the dialog box is automatically activated for features with optional elements.
 - "yes"-(Default) Button is activated automatically.
 - "no"-You select the button.
- "feature_create_auto_ok"-Determines whether the system activates the OK button in the dialog box automatically when the last required element is defined.

- "yes"-Button is activated automatically.
- "no"-(Default) You select the button.

Creating the Base Feature

The base feature is the first feature created in the definition of a part. It is the "working block" of material that is refined and modified until the final design. The displayed part axes are oriented such that the X-axis extends from left to right on the screen, the Y-axis extends from the bottom to the top of the screen, and the Z-axis points out of the screen towards you. Base feature extrusions are always extruded in the positive direction (out of the screen towards you).

How to create a base feature

1. Choose **Feature** from the PART menu.
2. Choose **Create** from the FEAT menu. **Solid**, **Datum**, **User Defined**, and **DesignerIn** are the only options available for creating a base feature.
3. Choose the desired option. If you choose **Solid**, choose **Protrusion**. The system then displays the SOLID OPTS menu, which has the options **Extrude**, **Revolve**, **Sweep**, **Blend**, **Use Quilt**, and **Advanced**. For information on how to create a protrusion, see [Protrusions](#).

Restrictions

Creating a base feature protrusion differs slightly from adding a protrusion to a part. For a base feature protrusion, there is no existing part geometry to which to relate the new feature. Therefore, some of the elements will not be available. The possible choices are as follows:

- *Extrusions*-The only available depth element is Blind.
- *Revolved protrusions*-You can specify angle attributes only.
- *Blends*-The possible blend type options are Parallel, Rotational, and General. Blends can use only the depth option Blind, and the only available attributes are Straight or Smooth and, for rotational blends, Open or Closed.

Creating Datums as the Base Feature

The following sections describe how to create datum planes, coordinate systems, and graphs as the base feature.

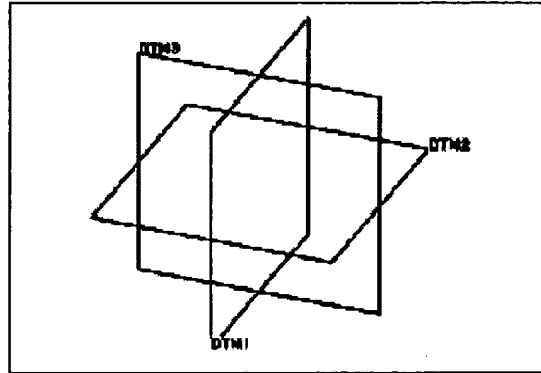
Datum Plane

You can create three orthogonal datum planes as the base feature before you add a solid feature. This is helpful when the first solid feature is going to be a sphere, toroid, or sculptured surface because it usually does not have the planar surfaces needed to reorient the model or specify sketching planes.

How to create a datum plane base feature

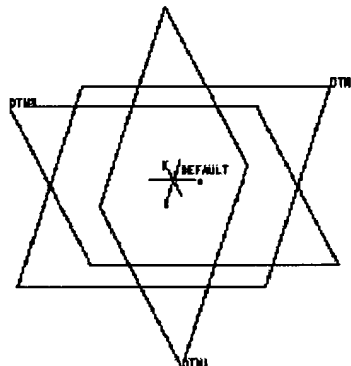
1. Choose **Feature** from the PART menu.
2. Choose **Create** from the FEAT menu, then **Datum**. Next, choose **Plane**.
3. The system displays the menu MENU DTM OPT, which has the following options:
 - **Default**-Create three orthogonal datum planes that intersect at the default origin. If you choose this option, three datum planes with the names *DTM1*, *DTM2*, and *DTM3* appear in the center of the screen at right angles to each other (see the following figure).

Default Datum Planes as the First Feature



- **Offset**-Create three orthogonal datum planes with specified offsets along the X-, Y-, and Z-axis. If you choose this option, the system prompts you for the offset value for the X direction, then repeats the prompt for the Y and Z directions. The system assumes that the X-axis is horizontal and pointing to the right, the Y-axis is vertical and pointing up, and the Z-axis is normal to the screen and pointing towards you. You can specify offset = 0. As soon as you have entered all three values, the system displays three datum planes with the names *DTM1*, *DTM2*, and *DTM3*. The system also displays a coordinate system named **DEFAULT**, which is located in the center of the screen (see the following figure).

Offset Datum Planes as the First Feature



Coordinate System

You can create the default coordinate system of the part as the first feature of the part.

How to create a coordinate system as the base feature

1. Choose **Create** from the FEAT menu, then **Datum**.
2. Choose **Coord Sys** from the DATUM menu. Pro/ENGINEER creates the coordinate system named *CS0*.

Graph

If you anticipate needing a graph for relations in your first feature, you can create a graph feature at the beginning of part creation. See [Graphs](#) for information on how to create a graph feature.

Creating a User-Defined Feature as the Base Feature

If you have the Pro/FEATURE module, you can create a UDF as the base feature (see [UDFs and Groups in Copying Features](#)), as long as it has no references to other features. For example, you might have a UDF that is a solid feature that references two datum planes, which reference the default coordinate system (created as the base feature). All the features must then be included in the UDF to become the base feature for a new part.

Adding Features

Once you have created the base feature, you can add other features (see [Construction Features](#)). The order of creating features is important. A feature will not "know" anything about the features that are created later. For example, if you create a hole using the Thru All option, then add a protrusion at its bottom, the hole will not penetrate the protrusion. You can change the order in which features are regenerated using the Reorder option in the Feat menu (see the section [Reordering Features](#) in the Chapter [Modifying the Part](#) for more information). Using this example, if you reordered the protrusion before the hole, the hole would then penetrate the protrusion.

For placement or dimensioning references, each feature depends on one or more of the earlier features and is considered to be the "child" of these features. This parent-child relationship is very important when you modify a part. When you suppress or delete a parent, the system asks what action to take for its children, including whether to suppress or delete them. Note that you cannot reorder features so the child is "earlier" in the feature list than the parent. However, you can modify parent-child relationships using the Reroute option (see [Rerouting Features](#)), or the Redefine and Scheme options (see [Redefining Features](#)) in the Feat menu.

Just as you would not ask a shop to weld on a piece of metal and then cut it off, you should *not* use "filler" features or features that negate other features. For instance, to fill a void made by a cut feature, rather than creating a protrusion that fills the void, just delete the cut. Similarly, to remove an entire protrusion, rather than creating a cut that removes the protrusion, simply delete the protrusion. Using fillers leads to excessively high numbers of features, slower regeneration time, and a more complex part. Make sure each feature accomplishes some goal in the overall design.

Creating the Initial Solid Feature

You add and locate the initial solid feature with respect to the base feature (one plane is used as a sketching plane and the other two are used for locating the cross-section). This follows the same

methodology as when you add a sketched feature.

How to create the initial feature after creating a datum

1. Choose **Create** from the FEAT menu, then **Solid**. The system displays the SOLID menu.
2. Choose **Protrusion** from the SOLID menu. Choose the desired form option, and **Solid** or **Thin** (available only with the Pro/FEATURE module), then choose **Done**.
3. Create the sections, as appropriate, locating them with respect to the base feature datums.

Dimensioning the Part

You dimension part features as you create them. There are two ways to dimension features:

- Create dimensions in Sketcher mode (see [Sketching on a Part](#)).
- Enter values for feature parameters as prompted. For example, when you create a "through" hole, you must enter values for diameter and placement dimensions.

Once the feature is created, you can display its dimensions at any time by choosing Modify and picking on the feature.

It is good practice to add relations immediately after feature creation (see [Relations](#) in the *Fundamentals* manual). This allows you to capture the design intent of your feature right away. For example, perhaps a slot should always be centered on a particular protrusion. If you write the relation to do this immediately after you create the slot, when a parameter affecting the protrusion is updated, you will retain the design intent of centering the slot.

Place dimensions driven by relations on *layers* (see [Using Layers](#)), then blank the layers. This helps you to understand which dimensions are really driving the model. Also, change the "dim symbol" to help further explain the function of important dimensions. For more information, see [Read-only Features](#)

Fractions

When you enter a fraction dimension, you can enter it in decimal format (1.125) or as a fraction equation (1+1/8).

To display dimensions as fractions, use the following configuration options:

- *create_fraction_dim*-Set to "yes" to display all the dimensions as fractions.
- *dim_fraction_denominator*-The only denominators allowed are 2, 4, 8, 16, 32, and 64. If you enter a decimal or fraction value that does not convert exactly to a fraction having one of these denominators, the value will be changed to the closest value. For example, entering "1+5/27" displays a value of "1-3/16".
- *use_major_units*-This option is not used when you are in Sketcher mode, but controls the display of

dimensions in the model. Set this to "yes" to display dimensions as feet-inches or meter-millimeter. For example, 25.75 can appear as 25-3/4 or 2' 1-3/4". Similarly, 1024 millimeters appears as 1024 or 1m24. Millimeters are never converted to fractions.

Once a dimension exists as a fraction or a decimal, you can modify the format of the dimension, as described in [Modifying the Part](#).

Reference Dimensions

You can create reference dimensions on existing part geometry. These dimensions have the suffix "REF", or are in parentheses when they are displayed. Reference dimensions are intended for information only. They have one-way associativity: you cannot modify them, but they reflect modifications made to the basic dimensions of the part. For more information, see [Reference Dimensions](#) in the *Fundamentals* manual.

Negative Dimensions Environment

Pro/ENGINEER allows you to enter negative dimension values for sketched sections. You set the mode of operating with negative dimensions using the configuration file option "show_dim_sign". The possible modes are as follows:

- *yes*-Operate in a true signed dimension environment. In Part and Assembly modes, section negative dimensions are displayed with the negative sign. Negative dimensions always display the sign in Sketcher mode. When you modify a dimension, you must enter its true signed value.
- *no*-Display all section dimensions in Part and Assembly modes as positive. This is the default value of the configuration file option. Modifying the value with a negative dimension causes the section geometry to reverse (if possible), even though its true signed value is negative (because it is displayed as positive).

Modifying the Part

There are several ways to modify existing part geometry:

- Modify the dimension values.
- Redefine existing features to change their attributes, section, or the direction of feature creation.
- Reorder features.
- Insert new features.
- Suppress or delete features.

Note:

When you suppress or delete features, you must reroute or modify the dimensioning scheme of their children, unless they are also to be suppressed or deleted.

For more information, see [Modifying the Part](#).

Using Layers

You can blank datum entities, coordinate systems, and dimensions from subsequent displays by adding them to a layer and choosing Blank from the Layer Disp menu.

Construction features (such as rounds) added to a layer are *not* affected when the layer is blanked. However, you can add them to a layer and then use the Suppress, Resume, or Delete option in the Feat menu to affect all these features. For more information on using layers, see [Layer Functions](#) in the *Fundamentals* manual.

Regenerating the Part

After you have modified a part, it must be regenerated to recalculate all the geometry affected by the changes and to update the display.

The system regenerates the part feature by feature, in order of creation or, if you used the Reorder command, in order of the current feature list. In most cases, regeneration starts with the earliest feature affected by the changes.

If the modifications you performed make it impossible for Pro/ENGINEER to recreate the part (as described in [Resolving Feature Failures](#)), you can choose between options that do the following:

- Allow you to perform several powerful recovery operations on the model.
- Request the system to try to recover the part.
- Allow you to investigate the cause of the regeneration failure.
- Redefine feature elements that caused the failure.

See [Regenerating the Part](#), for more information on part regeneration.



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